

## Overview

This document gives step-by-step instructions for simulating turbulent flow and heat transfer for a single heated block in a duct. To set up and solve this model with I-DEAS ESC the following new features are introduced.

1. Creation of new solid materials to specify properties for heat conduction.
2. Creation of a 3D mesh to model heat conduction within a solid that it is in contact with air.
3. Use of a flow blockage
4. Use of flow surfaces to define heat loads
5. Model partitioning for mesh control

## Physical Problem

Figure 1 gives the plan and elevation views of the geometry. A cubic block of dimension  $h$  is situated near the inlet of a rectangular duct of height  $H$  and width  $W$ . The fluid (air) entering the duct has a uniform velocity  $U_{in}$ , and temperature  $T_{in}$ . The dashed box shown in the plan view indicates a

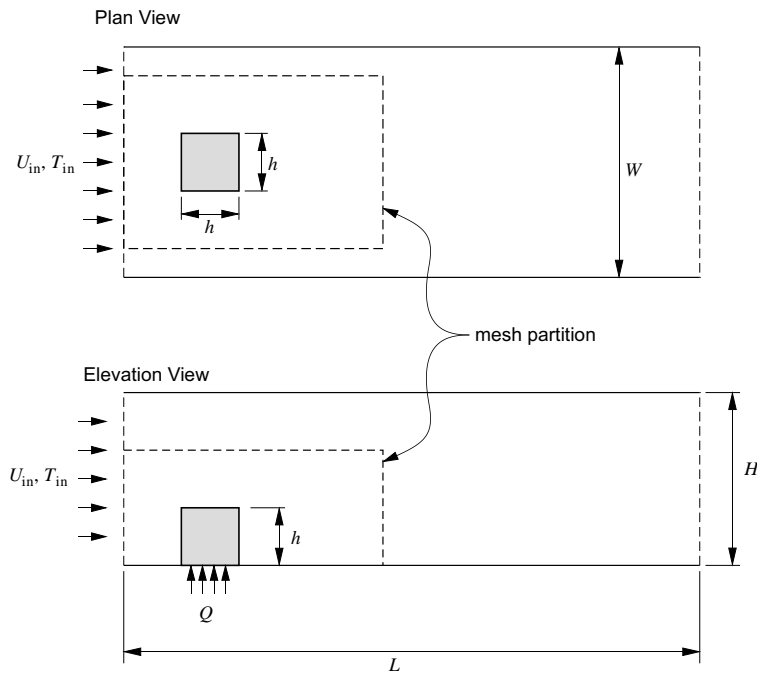


Figure 1: Geometry of the cubic block in the rectangular duct.

subvolume of fluid around the block that it used to control the mesh density near the block. This subvolume is created as an IDEAS partition, but does not exist as a physical boundary to the flow. A heat load  $Q$  is applied to the base of the block. This heat is conducted upward through the block and is transferred to the air by convection.

The geometrical parameters used in this cookbook are

$$h = 3 \text{ cm} \quad W = 15 \text{ cm} \quad H = 9 \text{ cm} \quad L = 30 \text{ cm}$$

The thermal and flow parameters are

$$U_{\text{in}} = 1 \text{ m/s} \quad T_{\text{in}} = 20^\circ\text{C} \quad Q = 10 \text{ W}$$

The hydraulic diameter of the duct is

$$D_h = \frac{4WH}{2(W+H)} = 10.29 \text{ cm}$$

The kinematic viscosity of air at  $20^\circ\text{C}$  is  $\nu = 1.51 \times 10^{-5} \text{ m}^2/\text{s}$ , so that the Reynolds number of the duct based on hydraulic diameter is

$$\text{Re}_{D_h} = \frac{U_{\text{in}} D_h}{\nu} = \frac{(1 \text{ m/s}) (0.1029 \text{ m})}{1.51 \times 10^{-5} \text{ m}^2/\text{s}} = 6815$$

Thus, the flow in the duct is turbulent

To set the pseudo-time step in the ESC solver<sup>1</sup>, we need to estimate the time it takes for a particle transit to move past the geometric features of interest. The ESC User's Manual recommends a time step of  $\Delta t = L_c/(2U_c)$ , where  $L_c$  is a characteristic length scale for the flow, and  $U_c$  is the characteristic velocity scale. For this problem,  $U_c$  is the inlet velocity  $U_{\text{in}}$ . Two choices of length scale are  $h$ , the size of the cube, or  $L$  the length of the duct. Thus, two candidates for the time step are

$$(\Delta t)_h = \frac{h}{2U_{\text{in}}} = 0.015 \text{ s} \quad (\Delta t)_L = \frac{L}{2U_{\text{in}}} = 0.15 \text{ s}$$

If the model converges with a pseudo-time step of  $(\Delta t)_L$ , then there is no need to use a smaller time step. If the model does not converge with  $\Delta t = (\Delta t)_L$ , then a smaller time step should be tried. A reasonable lower limit for the time step is 0.01 s.

---

<sup>1</sup>The pseudo-time step is only used to enhance stability. The flow is steady.

## Creating the Model

1. Launch I-DEAS, and open a new model file.
2. Switch to the *Master Modeler* task.
3. Select “mm (newton)” units.
4. Sketch a 300 mm  $\times$  120 mm rectangle on the work plane. (See Figure 1.) Fix the dimensions of this rectangle and extrude it a distance  $H = 90$  mm to create the fluid volume of the duct.
5. Select *Sketch on Face* (p1, r1, c1).
  - Select the bottom surface of the duct
  - Sketch the plan view of the cube. Dimension the square cube to be 30 mm on a side. Locate the cube 60 mm from the side walls (center it in the duct) and 30 mm from the inlet.
  - Select *Extrude* (p1, r5, c1)
    - ▷ Click the *right mouse button*, and select *partition* from the pop-up menu
    - ▷ On the *Extrude/Partition Create* form, select *distance* and enter 30 mm.
    - ▷ Check that *partition* is selected from the right-most pop-up menu
    - ▷ Click OK
6. Select *Sketch on Face* (p1, r1, c1).
  - Select the bottom surface of the duct
  - Sketch the plan view of the dashed rectangle in Figure 1. Dimension the rectangle to be 60 mm wide (across the duct) and at least 100 mm long (in the flow direction).
  - Select *Extrude* (p1, r5, c1)
    - ▷ Click the *right mouse button*, and select *partition* from the pop-up menu
    - ▷ On the *Extrude/Partition Create* form, select *distance* and enter 60 mm.
    - ▷ Check that *partition* is selected from the right-most pop-up menu
    - ▷ Click OK

At the completion of these steps you have a single part that is partitioned into three volumes, as depicted in Figure 2. The innermost volume is the cube. The cube is surrounded by a brick-shaped volume of air that will have a fine mesh. The remaining volume is the bulk of the air in the duct.

- Name the part

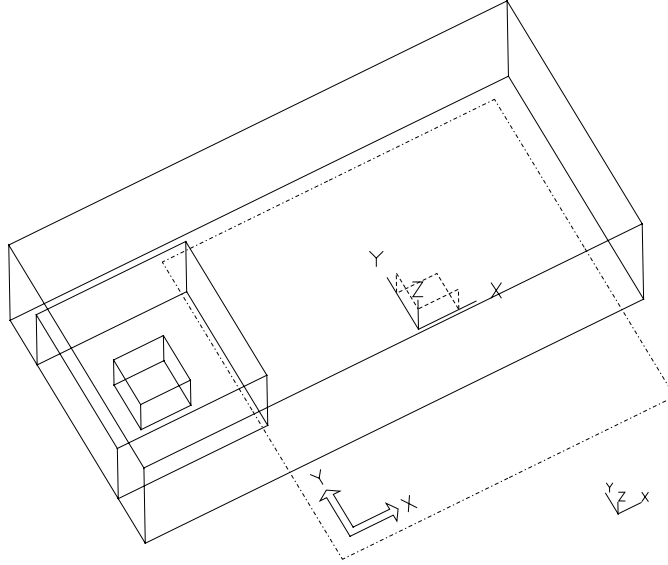


Figure 2: Partitions in the completed model.

## Meshing the Model

### Overview:

To mesh this part we will use a combination of mapped and free meshing. Mapped meshes are used

- on the bottom surface of the cube
- in the cube volume
- on the inner part of the inlet surface
- on the outlet surface

The mesh on the bottom surface of the cube is needed to apply the heat load boundary condition. The inner part of the inlet surface is the rectangular area formed by the upstream face of the fine mesh partition. The mapped meshes enable us to easily control the mesh density in the solid, and on faces of the air volume.

Free meshing is used

- on the outer part of the inlet surface
- in both subvolumes occupied by the air

The combination of the free mesh and mapped mesh on the inlet surface is shown in Figure 3.

Other meshing strategies could be used. For example, free meshes could be used on all surfaces and volumes.

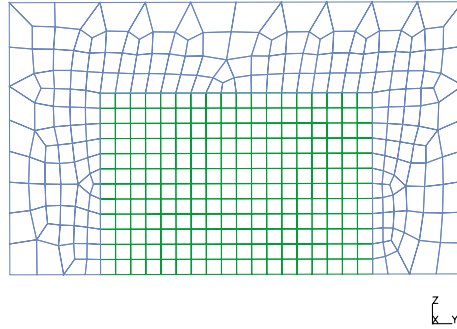


Figure 3: Combination of mapped mesh and free mesh on the inlet surface.

The major steps in meshing the model are:

- Prepare for meshing: name the FE model
- Define a *Null Shell* physical property
- Define a *TMG material* for aluminum for the block
- Define a non-conducting *TMG material* for surface meshes
- Apply surface meshes to domain boundaries
- Apply mapped volume mesh to the aluminum cube
- Define volume meshes for fluid in two steps. First mesh the inner fluid volume adjacent to the cube. Then mesh the outer fluid volume that fills the remainder of the duct.

Note that a surface mesh also needs to be defined on the surfaces of the cube in contact with the air. This surface mesh is used by the flow surface to define the thermal connection between the solid (aluminum) and the fluid (air). We will use the *flow blockage* feature of the ESC task to define the aluminum cube as a solid that is impermeable to the fluid. The *flow blockage* feature automatically coats the surface with a surface mesh.

#### Steps for Meshing the Model:

1. Switch to *Meshing* task (second drop-down menu in upper right hand corner).
2. Click *Create FE model* (p1, r6, c2)
  - Name the FE model
  - Set the default part material to be *ESC air*
  - Click OK
  - Click OK to dismiss the warning about air not being a suitable material for stress analysis.

3. Click *Material Property* (p1, r5, c1)  
 → *material selection* form opens
  - Click *Quick Create* (third button on the right side of the form)
    - ▷ Enter material name: “Aluminum 2024-T6”
    - ▷ Select *TMG Solid* from the drop-down menu
    - ▷ Select “Mass Density” from the list of material properties
      - Enter  $2.77\text{e-}6$  kg/mm<sup>3</sup>
      - Click *Modify Value*
    - ▷ Select “Thermal Conductivity” from the list of material properties
      - Enter 17700 mN mm/mm/C/sec
      - Click *Modify Value*
    - ▷ Select “Specific heat below phase change” from the list of material properties
      - Enter  $8.75\text{e-}8$  mN mm/kg/C
      - Click *Modify Value*
    - ▷ Click OK
  - Click *Quick Create* (third button on the right side of the form)
    - ▷ Enter material name: “Null Conductive”
    - ▷ Select *TMG Solid* from the drop-down menu
    - ▷ Select “Thermal Conductivity” from the list of material properties
      - Enter 0 mN mm/mm/C/sec
      - Click *Modify Value*
4. Apply *thin shell* mesh to the bottom of the cube. Use the *Null Shell* physical property, and the *Null Conductive* material property.
5. Apply mapped solid mesh to the cube. *Be sure* to select “Aluminum 2024-T6” as the material.
6. Mesh the inlet and outlet surfaces with the *Null Shell* physical property, and the *Null Conductive* material property.
7. Mesh volume with **ESC AIR** as the material.

## Applying Boundary Conditions

Boundary conditions for this problem are

- Heat load of 10 W on the bottom of the cube
- Fan inlet with a uniform velocity of 1 m/s at 20 °C
- Vent to the ambient at the outlet surfaces
- Adiabatic wall boundary conditions
- ESC *flow surfaces* on the cube walls

The flow surfaces on the cube walls is implemented with the *flow blockage* feature of ESC.

### Steps for Applying Boundary Conditions

1. Create the solid as a flow blockage
  - Click the *flow blockage* button (p1, r2, c3)
  - Select the cube volume (clicking on the bottom surface allows selection of just the cube volume)
    - ▷ Name the blockage “Aluminum block”
    - ▷ Verify that the blockage type is “solid”
    - ▷ Check “convection”
    - ▷ Click *create surface properties*
      - Name the surface property “smooth surface”
      - select *use surface roughness of*, and enter 0 mm
      - Click *convection heat transfer coefficient*. This opens the *Additional Auto Calculate Options* form
        - ◇ uncheck *Electronics Enclosure*
        - ◇ Click OK
      - Click OK
    - ▷ Click the ☐ button to the right of *Surface Properties*
      - Select *smooth surface*
      - Click OK
    - ▷ Click OK.
2. Apply the heat load to the cube: click *Thermal BC Create* (p1, r1, c2)
  - Select the bottom surface of the cube
  - Name the boundary condition “Block heat load”
  - Verify that *heat load* is selected
  - select W from the pull-down menu
  - enter a value of 2
  - Verify that the heat load is 2 W
  - click OK

3. Apply the fan boundary condition to the inlet: click *Fan* (p1, r3, c1)
  - Select *both* parts of the inlet surface
  - In the *Fan create* dialog box:
    - ▷ Change the name of the fan (“inlet fan” would be a good choice)
    - ▷ Select the *inlet* radio button
    - ▷ Set the flow parameter for the fan:
      - Select *velocity* from drop down menu
      - Enter 1 m/s.
    - ▷ Explore the options, but do not change them
    - ▷ Click OK.
4. Apply a vent boundary condition to the outlet.

## Prepare to Solve, and Solve the Model

1. Use standard procedures to set the solver options
  - KE model
  - Thermal solve
  - Set physical time step to 0.2 s
  - Set working directory:
    - ▷ Click *Study Setup* (p1, r4, c1)
    - ▷ Select the run directory
    - ▷ Click OK
2. Solve the model
3. View the results